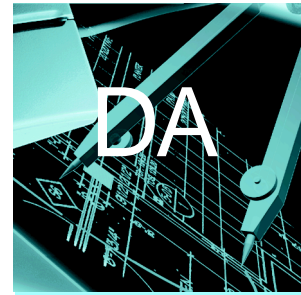


# Drafting Aids



The Design menu in the Mechanical Engineering Workspace contains several commands which are useful in the creation of mechanical drawings. Many of these commands can also be found in the Design Menu of other workspaces and any of the commands can be added to any workspace using the Workspace Editor, (refer to page 19-1, Customizing VectorWorks, in the MiniCAD VectorWorks User's Manual. This manual describes commands that aid in 2D drafting.

To use any command in this guide:

1. Select the option from the **Design Menu** in the **Mechanical Eng** workspace.
2. Enter the desired criteria.
3. Click **OK**.

For each tool, the path is listed. For example, Design>Drafting Aids>Break Line will access the Break Line command.

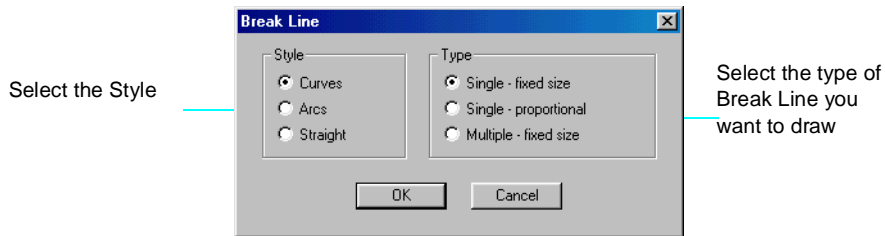
## CREATING BREAK LINES

### Design>Drafting Aids>Break Line

The Break Line command creates any of three break lines. The Single-proportional option draws the zig-zag lines in proportion to the overall length of the break line. In the single and multiple-fixed size break lines, the zig-zag lines are fixed in size regardless of the length of the lines. The break line can be drawn at any angle.

### In this Guide

- **Creating Break Lines**
- **Converting Arcs to Segments**
- **Cutting Plane Lines**
- **Dividing Lines into equal segments**
- **Creating drilled Holes and Tapped**
- **Creating Keyways**
- **Material Notes**
- **Section Notes**
- **Date Stamps**
- **File Name Stamps**
- **Drawing Border**
- **Creating Shaft Breaks**
- **Creating Slots**
- **Creating Arcs by Segment Length**
- **Creating Spirals**



Single - Fixed Size  
Style - Straight

A horizontal line with a single, sharp, V-shaped break in the middle.

Single - Proportional  
Style - Arcs

A horizontal line with a single, sharp, V-shaped break in the middle, where the break is formed by two intersecting arcs.

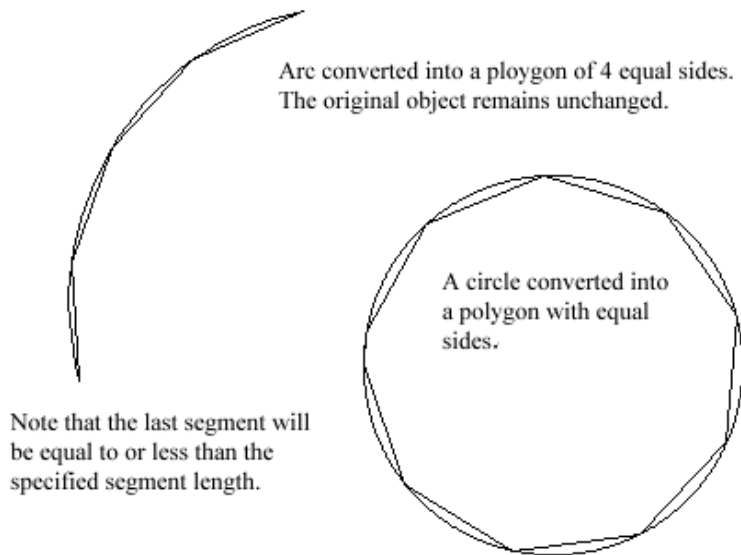
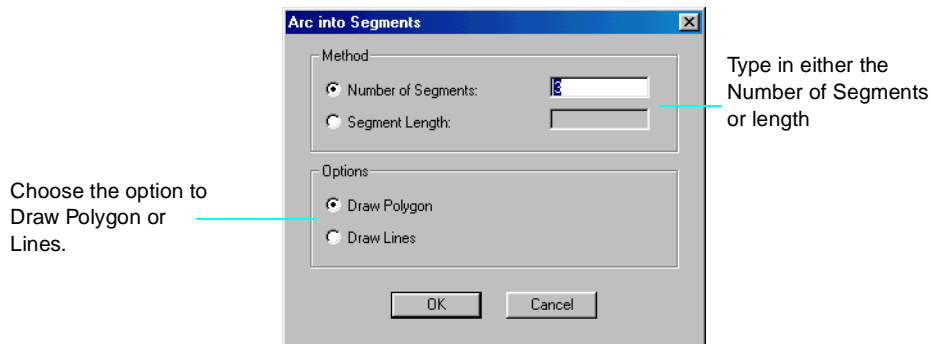
Multiple - Fixed Size  
Style - Curves

A horizontal line with two sharp, V-shaped breaks, one on the left and one on the right.

## CONVERTING ARCS TO SEGMENTS

### Design>Drafting Aids>Arc into Segments

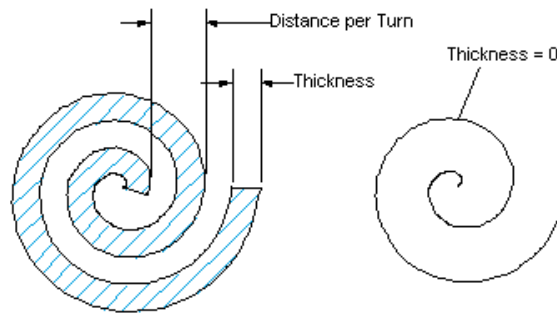
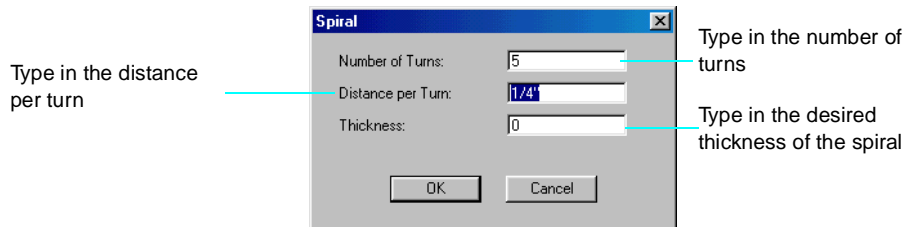
The Arc into Segments command will convert the selected arc or circle to an equal number of segments or will divide the arc or circle into segments of a given length. When the segment length option is chosen, the last segment will be equal to or less than the specified length. The segments can be drawn as lines or as a polygon.



## CREATING SPIRALS

### Design>Drafting Aids>Spiral

The Spiral command draws a spiral given the number of turns, the distance per turn, and the desired thickness.

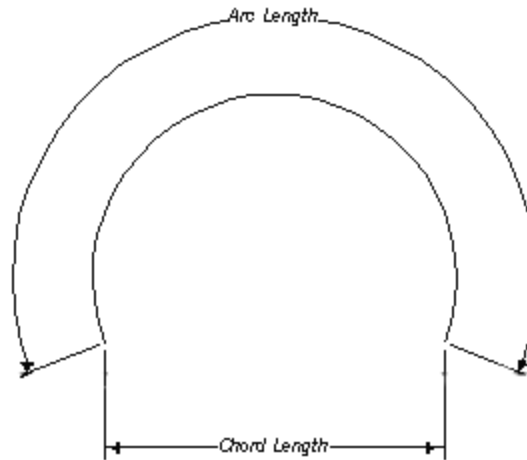
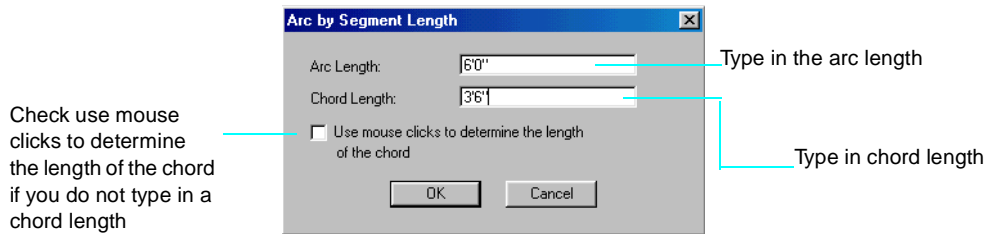


## CREATING ARCS BY SEGMENT LENGTH

### Design>Drafting Aids>Arc by Segment Length

The Arc by Segment Length command draws an arc with a given arc length and chord length. You can check "Use mouse clicks to determine the length of the chord", to manually draw the chord length.

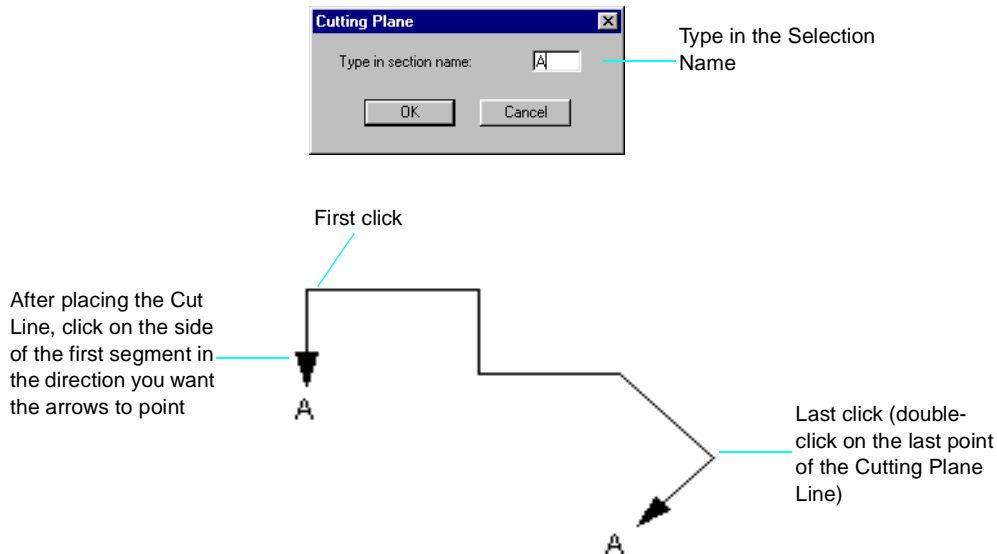
**Note:** When "Use mouse clicks to determine the length of the chord", is checked, if the length of the line is longer than the arc length, only a straight line will appear.



## CUTTING PLANE LINES

### Design>Drafting Aids>Cutting Plane

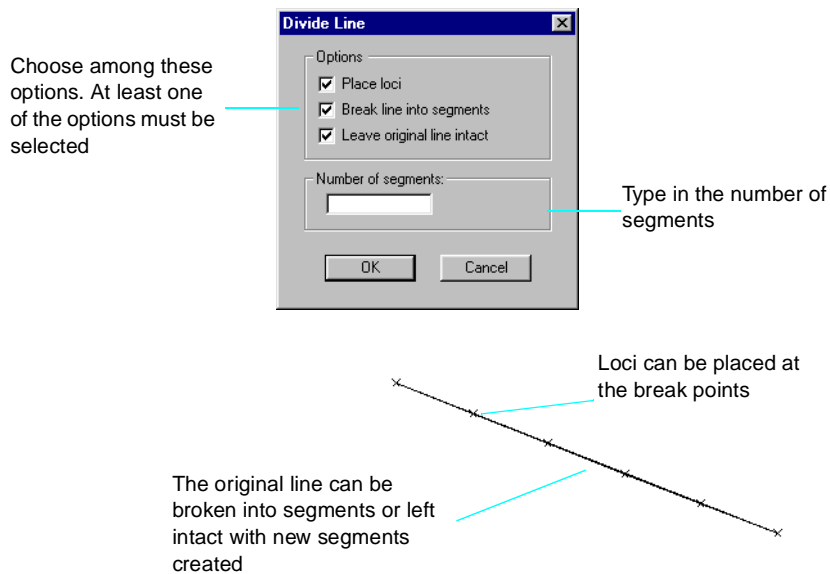
The Cutting Plane command draws a line or series of lines used to define a section cut. The line can have up to 10 segments and is defined by a series of mouse clicks. Double-clicking on the end point of the last segment ends the line definition. Click on the side of the first segment that you want the section arrows to appear to complete the command.



## DIVIDING LINES

### Design>Drafting Aids>Line into Segments

The Line into Segments command divides the selected line into any number of equal segments. The options are to place loci at the division points, to segment the existing line and/or to leave the original line intact.



## CREATING HOLES

### Design>Drafting Aids>Holes

The Holes command creates drilled and tapped holes with the option to draw a counterbore or countersink. The holes can be through holes or blind holes with a specified depth

..



Select either Drilled or Tapped Holes

Select the optional Counterbore, Countersink or none. If Counterbore is chosen, enter diameter and depth. If Countersink is chosen, enter diameter and included angle

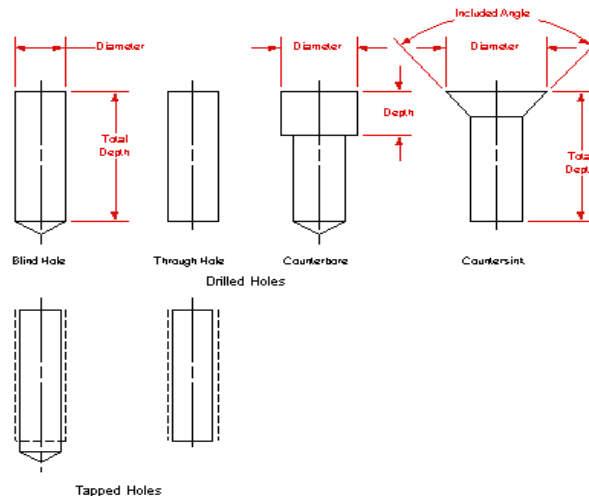
Select if a center mark is to be drawn

Select units either Inch or mm

Type in the Diameter and Total Depth (for side view)

Select the view and whether the hole is a through or blind hole

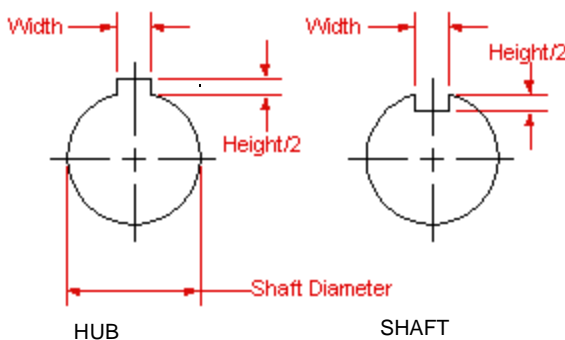
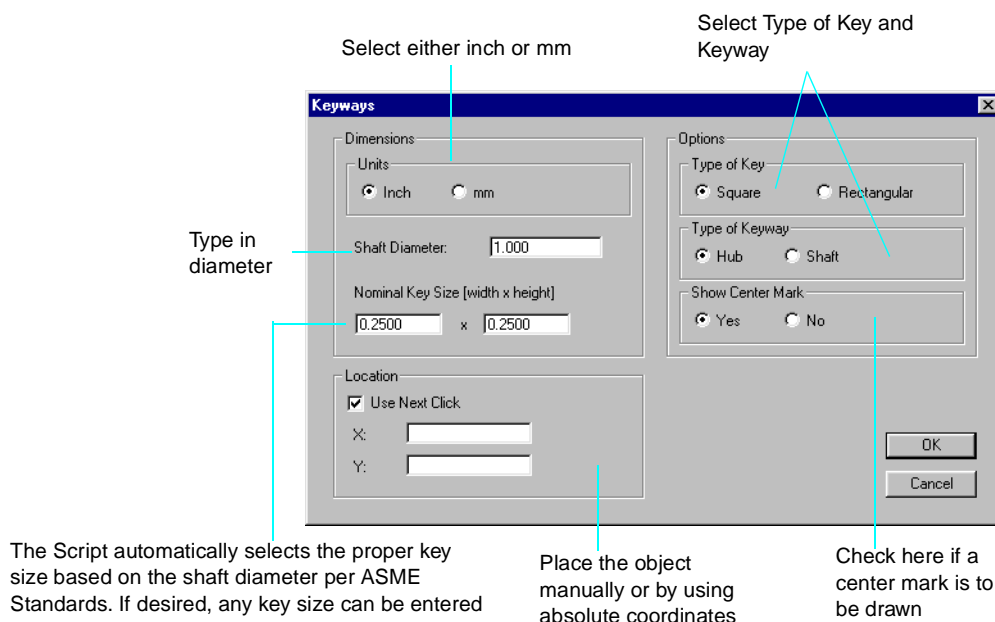
Place the hole manually or by using absolute coordinates



## CREATING KEYWAYS

### Design>Drafting Aids>Keyways

The Keyways command creates either an external (Hub) or internal (Shaft) Keyway and the corresponding shaft or hub. The standard ASME key size is automatically selected for the given diameter, but can be overridden by the user. A center mark can also be drawn.

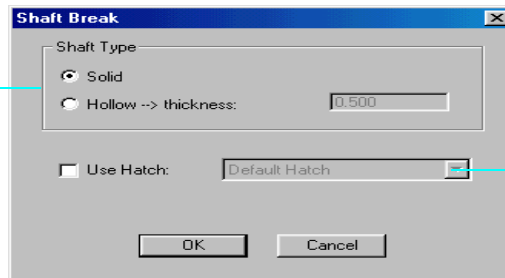


## CREATING SHAFT BREAKS

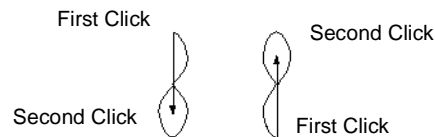
### Design>Drafting Aids>Shaft Break

The Shaft Break command creates a shaft break as shown. Click the two endpoints to create Shaft Break and double-click to end the script.

Select Shaft Type. If Hollow is chosen enter wall thickness

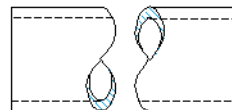
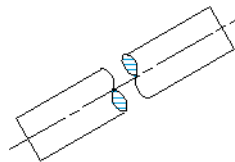


Select if the Shaft Break is to be hatched, and select the desired hatch



The orientation of the break is determined by the direction of the two clicks.

Solid Shaft Break



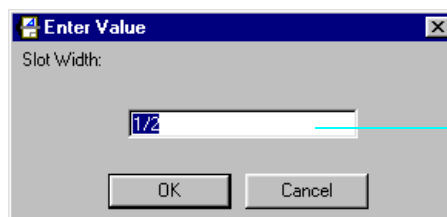
Hollow Shaft Break

The shaft breaks can be drawn on any angle.

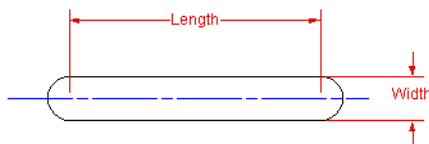
## CREATING SLOTS

### Design>Drafting Aids>Slots

The Slots command draws a slot as shown. The length of the slot is determined by two mouse clicks. Click center points of slot to create and double click to quit. The slots can be drawn on any angle.



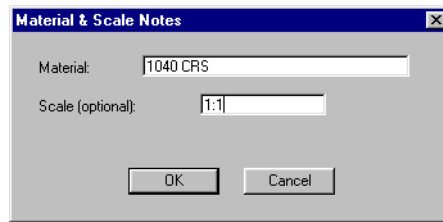
Type in the width of the slot



## MATERIAL NOTES

### Design>Drafting Aids>Material Note

The Material Note command places a note containing the material of the drawn part, and optionally the drawing scale anywhere on a drawing.



A dialog box titled "Material & Scale Notes" with a close button (X) in the top right corner. It contains two text input fields: "Material:" with the value "1040 CRS" and "Scale (optional):" with the value "1:1". At the bottom are "OK" and "Cancel" buttons.

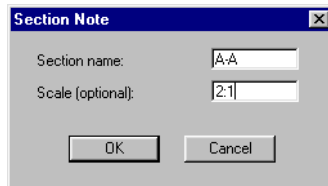
MATERIAL - 1040 CRS  
SCALE - 1:1

**Note:** The Material Note script uses the default text font, size and style. The scale note is 80% of the default text size.

## SECTION NOTES

### Design>Drafting Aids>Section Note

The Section Note script places a note defining a section and optionally the drawing scale anywhere on a drawing.



A dialog box titled "Section Note" with a close button (X) in the top right corner. It contains two text input fields: "Section name:" with the value "A-A" and "Scale (optional):" with the value "2:1". At the bottom are "OK" and "Cancel" buttons.

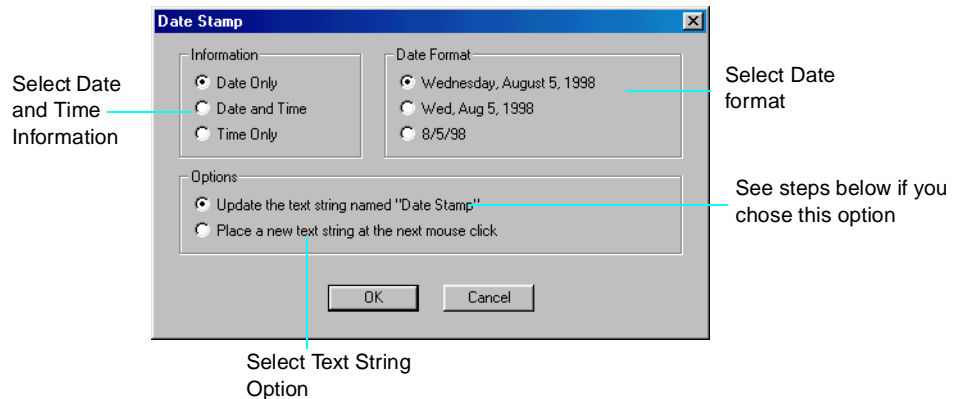
SECTION A-A  
SCALE - 2:1

**Note:** The Section Note script uses the default text font, size and style. The scale note is 80% of the default text size and the section name is 1.4 times the default text size.

## CREATING DATE STAMPS

### Design>Drafting Aids>Date Stamp

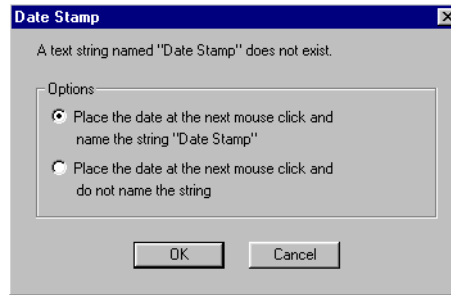
The Date Stamp command places the current date and/or time on the drawing. If there is a text string named "Date Stamp", the command will change the text string to the current date and/or time.



To create a text string named Date Stamp.

1. Using the text tool on the 2D Tool Palette create a text block.  
Highlight the text block.
2. Select **Object Info** from the Palettes menu if the Object Info Palette is not already opened.  
In the Object Info window select the Data tab and type Date Stamp in the first text box. It is this step that names the text string.
3. From the Design menu select Drafting Aids and then **Date Stamp**.
4. Select Update Text String Named Date Stamp in the Date Stamp dialog.
5. Click **OK**.

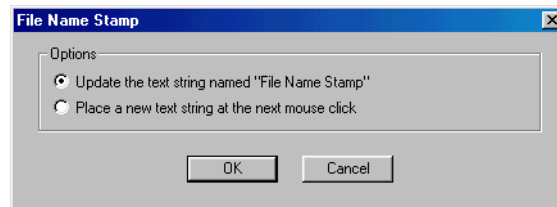
If you have not created a text string named Date Stamp, VectorWorks will do it for you. After you have selected the desired criteria in the Date Stamp dialog box, click OK. A new dialog box will appear prompting you to select the option to place the date at the next mouse click and name the string "Date Stamp" or place the date at the next mouse click and do not name the string.



## FILE NAME STAMP

### Design>Drafting Aids>File Name Stamp

The File Name Stamp command places the current file name on the drawing. If there is a text string named "File Name Stamp" on the drawing, the program will change the text string to the current file name.



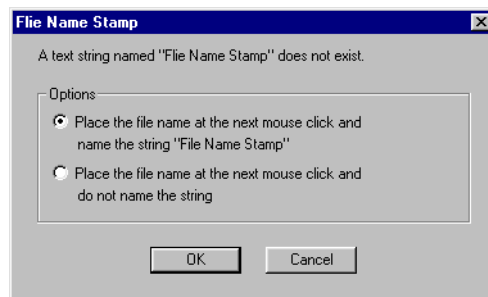
To create a text string named File Name Stamp

1. Using the text tool on the 2D Tool Palette create a text box.  
Highlight the text box.
2. Select **Object Info** from the Palettes menu if the Object Info Palette is not already opened.

In the Object Info window select the Data tab and type File Name Stamp in the first text box. It is this step that names the text string.

3. From the Design menu select Drafting Aids and then **File Name Stamp**.
4. Select Update Text String Named File Name Stamp in your File Name Stamp command.
5. Click **OK**.

If you have not created a text string named File Name Stamp, VectorWorks will do it for you. After you have selected the desired criteria in the File Name Stamp dialog box, click OK. A new dialog box will appear prompting you to select the option to place the date at the next mouse click and name the string "File Name Stamp" or place the date at the next mouse click and do not name the string.

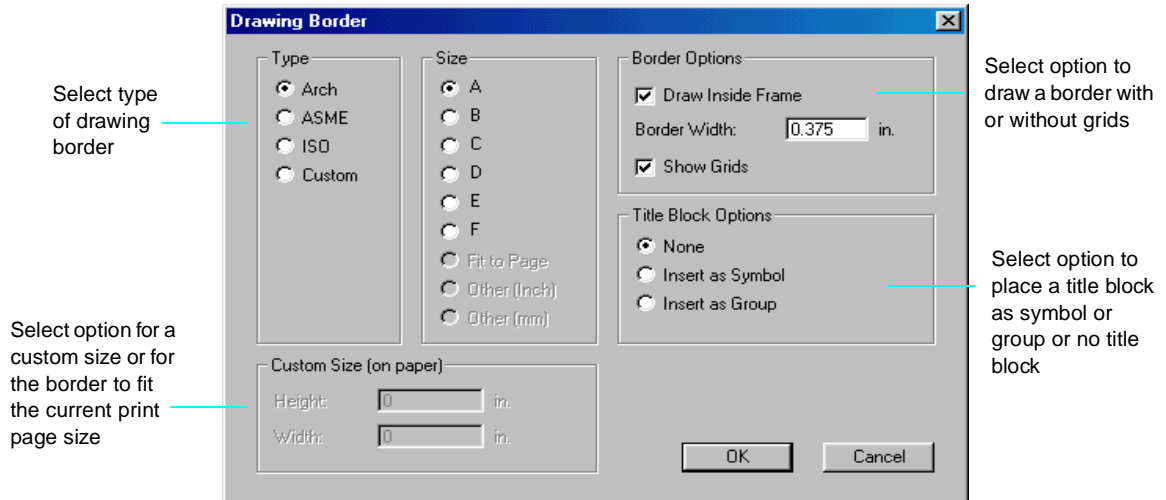


## DRAWING BORDER

### Design>Drafting Aids>Drawing Border

The Drawing Border command invokes a Drawing Forms window which draws a standard ASME, ISO, Architectural or Custom size drawing border. The drawing border is created on a layer named Drawing Border which, if it does not already exist, is created by the script. This new layer has a layer scale of 1:1.



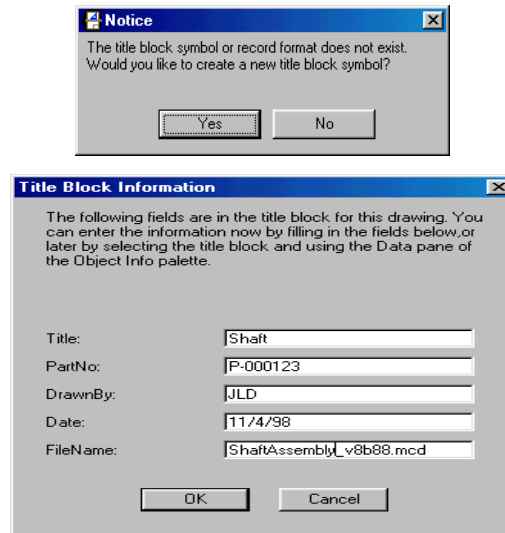


**Note:** If you want the drawing border layer to be a scale other than 1:1, after the border is drawn, change the scale of the layer to the desired scale. Select the drawing border and use the Scale Objects command to resize the drawing border.

**Note:** If the title block is a symbol it will not rescale. You must edit the symbol to resize it.

The command has the option to put a title block on the form. The title block is inserted as a symbol named "TitleBlock", which must be in the symbol library of the active drawing. The text fields in the title block are linked to a record format named "TitleBlockInfo", which also must be located in the current drawing. If the symbol and record format are not in the drawing at run time, the script will create them for you.

You can use your own title block and record format with up to 20 attached fields. The only restriction is that the title block symbol must be named "TitleBlock" and the record format must be named "TitleBlockInfo". The Drawing Border command will use your title block as long as these conditions are met.



When a title block option is selected, the Title Block Information window appears showing the fields in the record format TitleBlockInfo. Make desired edits or additions.

**Note:** If you need to edit a title block that is a symbol at a later date you can enter or change the text in the title block using the Data tab of the Object Info palette. If the title block was created as a group, you must ungroup the title block and change the text manually.